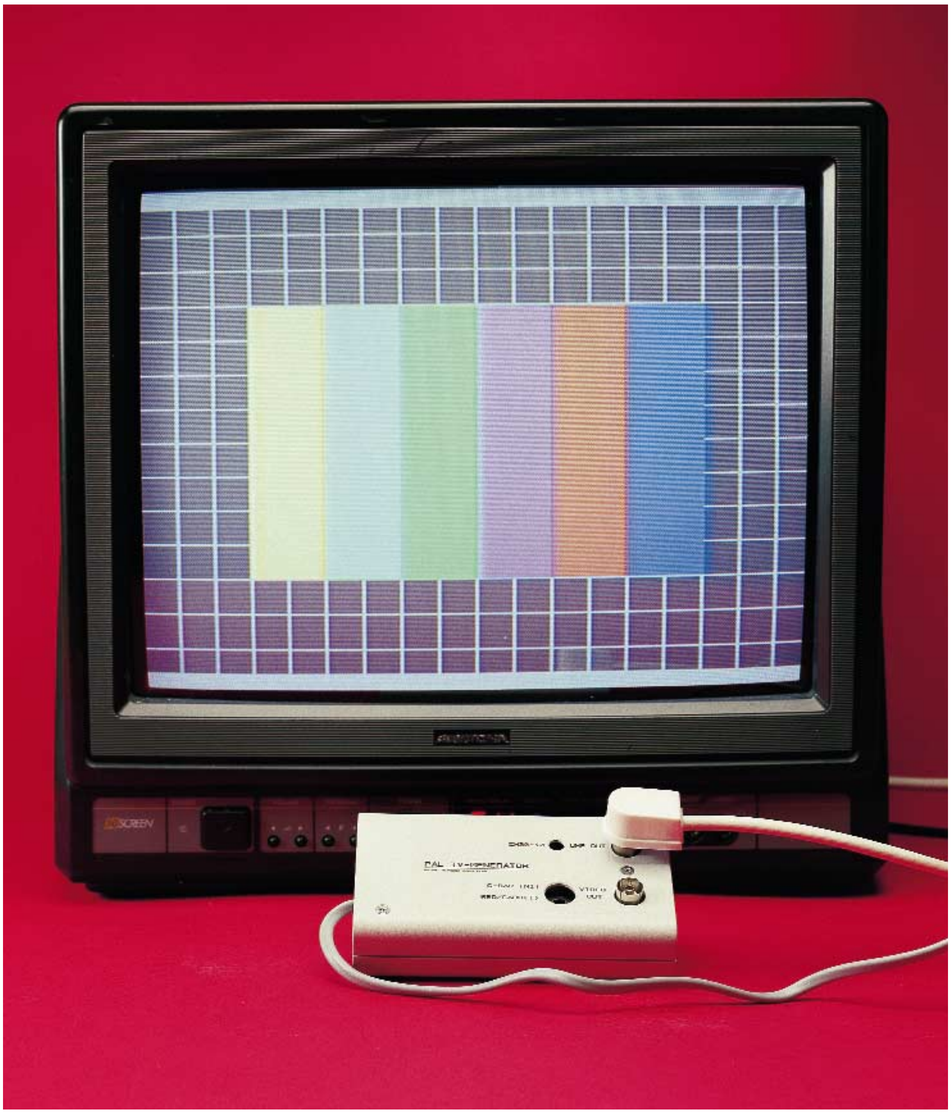
PAL TV Pattern Generator

prototype board layout using a PC

Design by W. Foede

Email: foede.koeln@gmx.de

In spite of digital television, an analogue test pattern generator is still part of the basic equipment of every electronics technician who works with television and video signals.



This device is an extension of the PAL Test Pattern Generator published in the January 1993 issue of *Elektor Electronics*. The GAL and the two counter ICs that were used at that time have been replaced by an EPLD with 64 I/O cells. The generator produces two test patterns: a colour bar pattern with a grid and a red microgrid. These are adequate for checking all important settings, such as picture position, picture geometry, convergence, grey balance, colour purity, colour and focus. In contrast to the original project, the generator described here includes a UHF modulator, so that the test signal can be fed to the television set not only via the video input sockets, but also via the antenna socket.

The subject of this article is not just the TV signal generator itself. A large part of the assembly instructions is devoted to the development and construction of the circuit. This is because a specific printed circuit board layout has not been developed for this article. Instead, we describe how the circuit can be built on a piece of perforated prototyping board (Veroboard) with a matrix of solder pads. Both the author of this article and the *Elektor Electronics* laboratory staff build most of their (initial) prototypes on such boards. This is because prototyping board

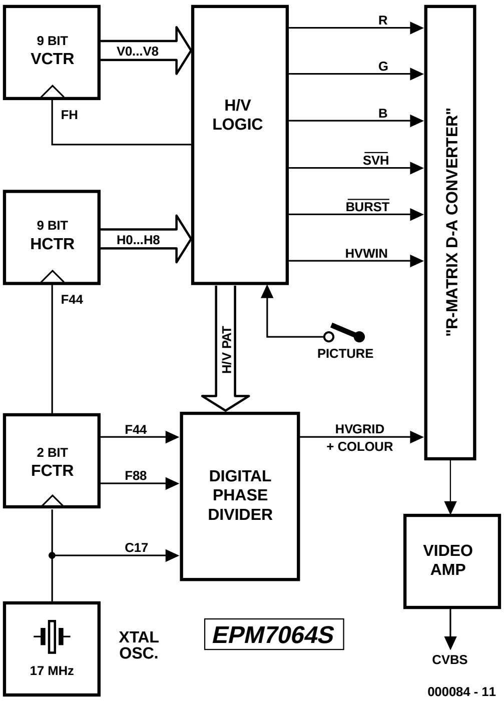


Figure 1. Internal block diagram of the EPLD.

assemblies are actually better than their reputation: they are easy to understand and easy to modify, the components can be inserted quickly, and the preparation of the board does not create any rubbish or additional effort.

There is also no reason why a circuit built on a prototyping board cannot be 'routed' using a PC. For many years, the author has been using a DOS-based, freeware CAD program for this purpose. The procedure will be described in detail later on, but first we provide an appropriately brief description of the circuit of the PAL TV Pattern Generator (since it works in the same way as the extensively documented version of 1993).

The logic: an EPLD

The PAL TV pattern generator consists of only a few components. Apart from the oscillator and video amplifier assemblies, which are built using discrete components, as well as the modulator, all of its major functions are housed in a single programmable component, which is an EPLD. The block diagram in **Figure 1** shows the internal structure of the EPLD, as well as the connections for the oscillator and video amplifier. The operating principles can be easily explained using this diagram.

The discrete quartz crystal oscillator supplies the common clock C17 for all signals. It runs at four times the frequency of the colour subcarrier (CSC) (4.43361875 MHz × 4 = 17.734475 MHz). C17 clocks the 2-bit counter FCTR, which in turn generates the signals F88 (approximately 8.8 MHz) and F44 (approximately 4.4 MHz). If F44 is divided by 284 using the 9-bit counter HCTR, the result is the horizontal frequency FH. This means that the generator works with a full-line offset, in contrast to the PAL standard, which prescribes a relationship of FH = FHT ÷ 283.7516 = 15.625 kHz (quarter-line offset). If the oscillator is tuned to exactly 4 × FHT, the horizontal line rate is here FH = 15.611 kHz.

The vertical frequency FZ (50.036 Hz) is obtained from the horizontal frequency via the 9-bit divider VCTR. This yields 624 lines for a full frame and no interlacing, which is called 'progressive' operation. This is good for test pattern images, since a stable image can be observed without any stress. It does not have the flickering edges of a normal television picture, which are only eliminated in top-end equipment using 100-Hz technology and full-frame memories. The small deviations from the standard are not visible in the rest of the image, and can be completely ignored. In the H/V logic, all of the picture, blanking and synchronisation signals are digitally programmed using the two 9-bit counters and logical operations, with precise timing and TTL levels.

Special effort is necessary to generate the colours. For each colour, the CSC has different phase states (hues) and amplitudes (saturation). In line with the PAL standard, the phases for each colour alternate from line to line. This also applies to the colour synchronisation signal (colour burst) on the back porch following each horizontal sync pulse. For six colours and the colour burst, a total of 14 phases are needed. Using the principle indicated in **Figure 2**, it is theoretically possible to digitally generate all the phases in a 45-degree matrix. By selectively inverting F44 and F88, as well as clocking with the rising or falling edge of C17, there are even several options for each phase. In addition, the propagation times of the individual signals and

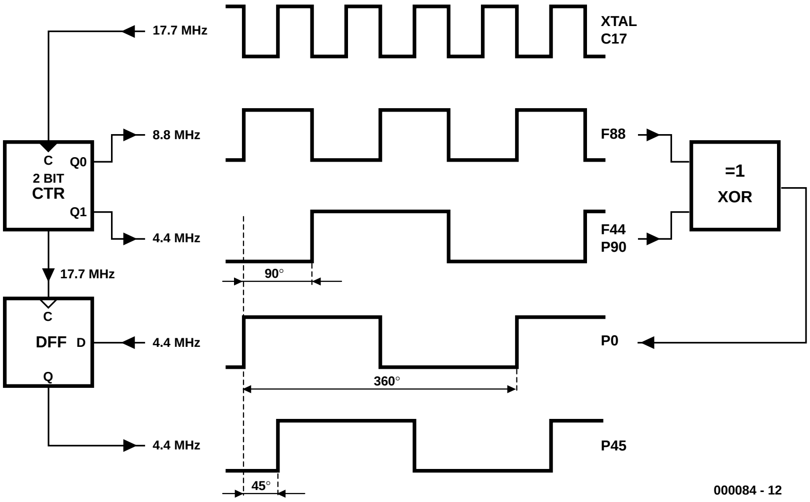


Figure 2. The working principle of the digital 45° phase splitter.

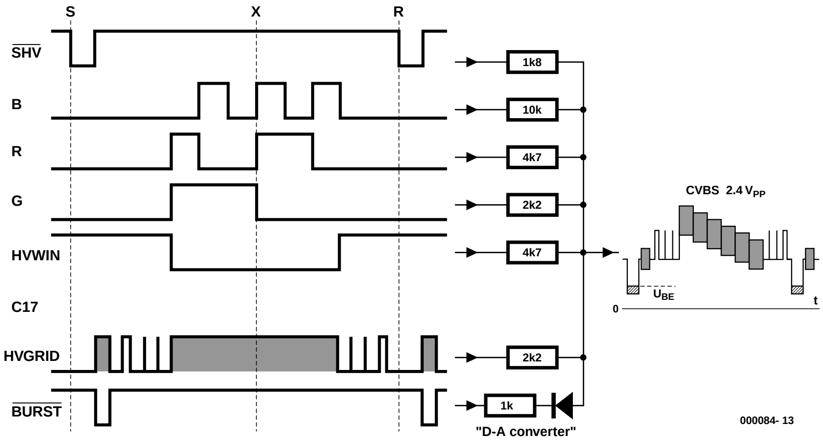


Figure 3. A D/A converter made from discrete components gives form to the composite video signal.

the duty factor of C17 affect the results. The phase states that most closely match the PAL standard were empirically selected using a PAL vectorscope; that is, by repeated trial and error. The result is a visually very usable colour bar pattern in test pattern 1. The timing relationship between the colour signal (C) and the picture signal (Y) is also provided by the H/V logic. The resistor network R1/R5–R9 represents a digital-to-analogue converter that, for each relationship, combines the digital levels of the individual components to produce the composite video signal (see **Figure 3**). HVWIN adjusts the brightness of the colour bars to match the rest of the image content, and BURST adjusts the amplitude and position of the colour burst on the back porch.

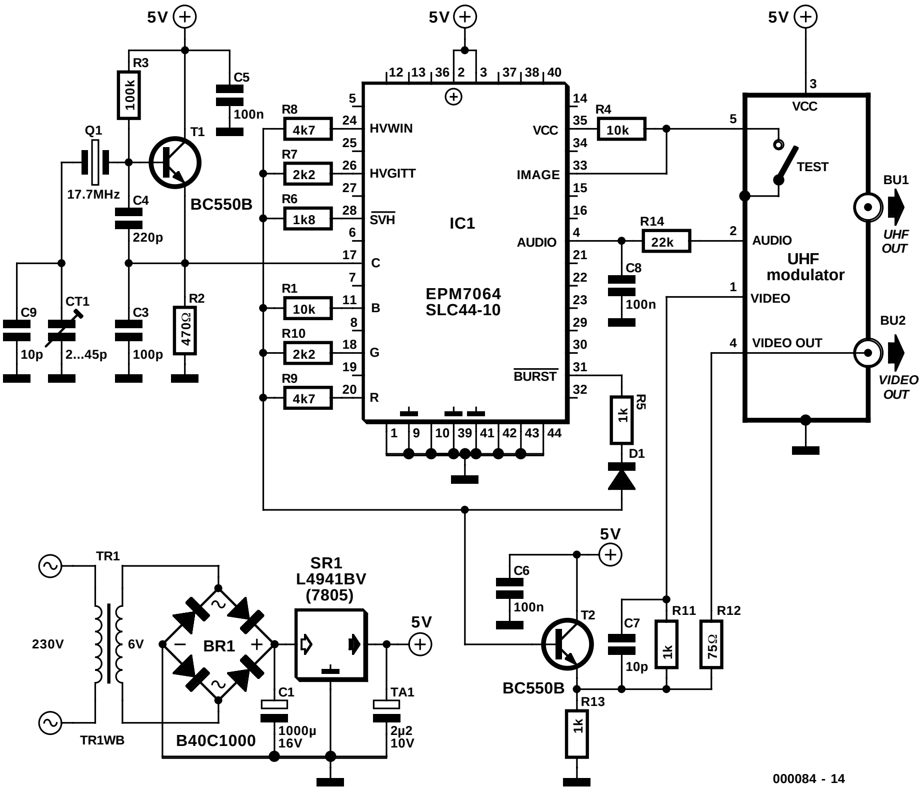


Figure 4. The EPLD needs only an oscillator and a buffer stage. The UHF modulator generates an HF signal, but in addition to an audio signal, it also provides the unmodified composite video image.

Before and after the logic

Let's take another look at the assemblies outside the ELPD in **Figure 4**. The clock generator, consisting of the overtone crystal X1 with the resonance capacitors C9 and CT1, is buffered by a transistor stage with T1, so that the clock signal C17 can drive a reasonable load.

A video amplifier based on T2 is connected to the output of the EPLD. It provides a low-impedance output. The commercial UHF modulator is driven by the composite video signal (with frequency compensation provided by R14/C8) and by a heavily integrated square-wave signal from VCTR at around 500 Hz, which serves as the audio signal. In order to minimise the number of external components that have to be connected to the circuit board, the modulator is slightly modified so that its antenna input also serves as a video output and the test switch serves as a test pattern selector switch. The UHF antenna output of the modulator is also available.

The modulator contains a wideband amplifier with input sockets, for feeding through the antenna signal. This amplifier is not used, so the Vcc connection (pin 5) and the antenna socket are free. Pin 4 is either grounded or left open. The modulator is grounded via its case, so pin 4 can be made free, possibly after cutting an internal connection to ground. In this case, be sure to restore the ground connection inside the modulator with as short a lead as possible.

In order to modify the modulator as shown in **Figure 5**, first open the back cover of the modulator. The inner contact of the HF socket is always connected to an air-core coil and an SMD capacitor. Break this connection, and then pull out the free end of the coil and cut it off. Next, unsolder the capacitor. Now all that is missing is a wire to pin 4, which receives the composite video signal via R12. We can do without the test signal generated inside the modulator, which consists of two black vertical stripes and horizontal (but not vertical) sync pulses. The modulator works in normal mode when the contact point of the test switch is connected to ground. In the best case, one of the three contacts is grounded. When the switch is operated, the middle contact is connected to this grounded contact, and the third contact is free. If the two contact points are now bridged with a bit of solder, the free contact will be connected to ground when the switch is operated. After cutting the Vcc track to pin 5 inside the modulator, the switch contact that has been made free can be connected to pin 5. The SMD filter capacitor that is usually connected to pin 5 can be left in place, since the signal BILD is in principle a DC voltage. Before closing up the modulator, it is strongly recommended to carefully check the modifications with an ohmmeter.

Unfortunately, in some modulators the connections to the test switch are different, even though is serves the same purpose. Naturally, if you have any doubts about making the modifications, it may be better to use an external switch or an external video socket.

The modulator is soldered directly on top of the circuit board using four 1-mm solder pins. The electrical connections are extended using five solder pins.

From painful practical experience, we know that every circuit, no matter how nice, also needs an enclosure, and we must not overlook the power supply. The snap-together shell enclosure used here is very inexpensive and should be a stock item in every electronics shop. It consists of two half shells that are 15 and 25 mm deep, respectively, and it restricts the size of the circuit board to 117 × 61 mm.

The power supply consists of only a few components, so it can also be mounted on the main circuit board. This makes a mains adapter unnecessary. Any printed-circuit-board transformer with a capacity between 1 and 2.5 VA and suitable dimensions can be used. To be on the safe side, we have chosen a 5-V low-drop voltage regulator, but from 1.5 VA upwards a 7805 in a TO220 package can also be used.

Alignment

Before installing the EPLD, you should measure Vcc (+5 V) on pins 2 and 3 of IC1, on the collectors of T1 and T2 and on pin 3 of the modulator. In addition, it is recommended to use a continuity tester to check all ground connections (with no power applied to the circuit). You can verify C17 using an oscilloscope. The crystal should oscillate, and with a frequency counter connected to R2, you can take this opportunity to adjust C17 to 17.734475 MHz using the trimmer capacitor CT1.

With the power off, you can now install IC1. On the unloaded video output, you should find a composite video signal with a level of around 2 V_{pp}. The trimmer in the modulator can be adjusted to an UHF channel between 30 and 40. If you do not have a frequency counter available, adjust CT1 for the best colour reproduction on a television set. The test pattern with the red microgrid is suitable for checking and adjusting the focus with the colour contrast set to zero and increased black-and-white contrast. With low black-and-white contrast and maximum colour contrast, it can be used for testing colour purity. The main test pattern, with colour bars and a grid pattern, can be used for all other adjustments. When the picture position and geometry are properly adjusted, the broad white borders should fall just outside of the visible image area.

From concept to prototype

The development of a private project can be concisely described in the following manner. First you have an idea, and following this you sketch a design on paper and then construct a prototype, either in the form of a ‘component sculpture’ or using a socket-strip prototyping board. After a while, the circuit will actually work. Experienced constructors use the schematic editor of a layout program to prepare the schematic diagram, and then use the layout editor to produce the circuit board layout. The exposure mask is then glued and exposed or printed, and the circuit board is etched, drilled, stuffed and soldered. This is a lot of effort for a circuit board that in many cases still has bugs, especially if you do not do it all that often.

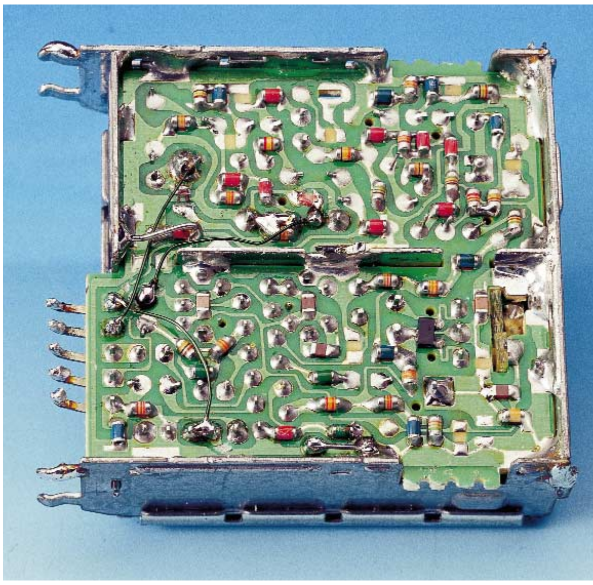


Figure 5. Modifications to the modulator (shown with the cover removed).

Most of the circuits published by the author over the years have been built in the demonstration stage using perforated prototyping boards with a matrix of solder pads, according to the method described below. The advantages are no chemicals, no broken 0.8-mm drills, standard materials and no waiting. If you work reasonably carefully, the results are certainly quite presentable. Surprisingly enough, the final *Elektor Electronics* circuit boards are often no smaller than the prototype. A well-built circuit on a perforated prototyping board can also be very helpful when making the conversion to a ‘real’ circuit board; it is in a manner of speaking half the battle.

Prototype layout with a program

The best way to achieve an elegant arrangement of the components on a perforated prototyping board is to use a layout program to generate the schematic diagram and the layout. If possible, this program should cost nothing and still be legal — in other words, it should be freeware. Two possibilities are *Eagle light* from CADSOFT and *Target 2001!V8 discover* from Ing.-Büro Friedrich, both of which have Windows interfaces and are available via the Internet (see ‘On the Internet’). For this circuit, the author has chosen *Easytrax,* which is a proven, free DOS program from Protel. It is especially suitable for beginners, and the included functions are fully adequate.

The program package consists of three zip archives, which together amount to less than 1 MB (oh, for the blessed days of DOS!). They can be obtained from the Free Downloads page on the *Elektor Electronics* website (select November 2000 items).

EASYTRAX.ZIP

The file EASYTRAX.ZIP contains the PCB drawing program EASYEDIT, the plotter output program EASYPLOT with drivers, BOM (bill of materials) and several input and output converters for other Protel programs. EASYTRAX.ZIP is initially unpacked onto a diskette (but not automatically installed). Following this, you can start the installation routine with a mouse click, and the drawing program and its accessories will be stored in the predefined folder C:\EASYTRAX. During the installation, you must select a video driver; standard VGA with 640 × 480 pixels will always work. You can try out other drivers that look like they may be usable. The video driver is installed in a separate folder called C:\GRAPHDRV. A separate DOS driver must be loaded for the mouse. An English-language manual of around 25 pages is located in the READ.ME file. An extended component library is available on the Internet, as well as extensive installation instructions (in German).

The drawing program, which can be used to draw a schematic diagram or a layout according to the component library used, is started using EASYEDIT.EXE. This program is quite easy to use, so we do not need to provide any instructions here. Everything that you need to know can be found in the ReadMe file.

Generating suitable component libraries is unfortunately difficult, but it is nevertheless sometimes necessary to do this preparatory work. You have to draw each component on a suitable layout level (such as OVERLAY) with its outline and solder pads (PAD), and then define it as a BLOCK. The cursor position when the left button of the mouse is clicked the first time after the block has been finished defines the reference point, which determines where the component appears with respect to the cursor after being called up from the library. The reference point and the pads should be specified using a 100-mil (2.54-mm) reference grid. The component is stored using the hotkeys L/A/Y. Components from the library can be converted back into editable individual elements (PRIMITIVE) by pressing the L/P keys and clicking on the component, after which the elements can be modified. The diameter of a circle can be adjusted using the left and right cursor keys (snapping to the reference grid), and a circle can be divided into quadrants by clicking on it and then pressing the cursor keys. A new function can be called up only after the current one has been stopped by pressing <ESC> once or twice. It is always possible to change the reference grid to 100 mil (2.54 mm) for this project using G <CR>, in order to place components (PLACE COMPONENT) or lay down tracks (PLACE TRACK). Similarly, +/– zoom (PgUp/PgDn) and refresh (F9) are always available. The holes of the circuit board are located on a 2.54-mm grid, or better put, on grid with a spacing of 100/1000 of an inch, or 100 mil. It is a good idea to set the default units to inches (IMPERIAL). Although this takes some getting used to if you are accustomed to working in metric, it makes sense, since almost all standard electronic components are based on a 100-mil grid. It's also easier to divide 100 mil by 2 or 4 than to divide 2.54 mm!

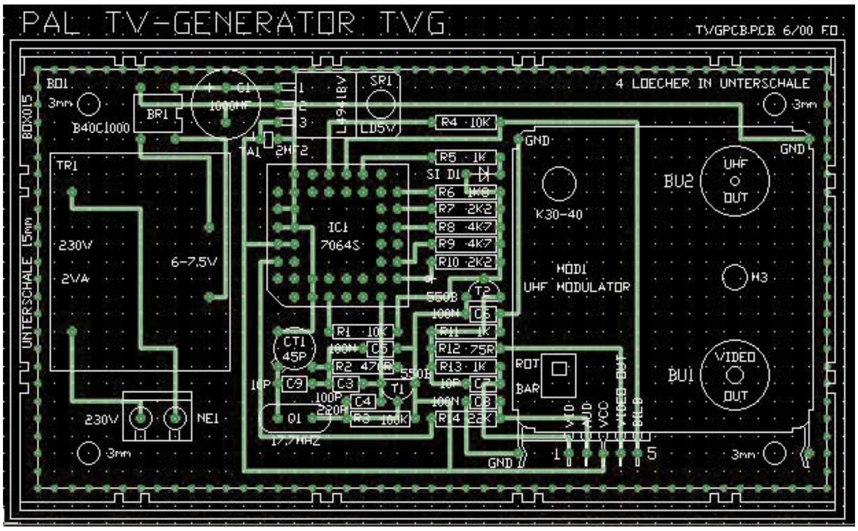


Figure 6. Screen dump of all circuit board layers under *Easytrax*.

The circuit board uses only four of the available layout layers (LAYER), namely OVERLAY for COMPONENT and STRING, BOTTOM (solder side)

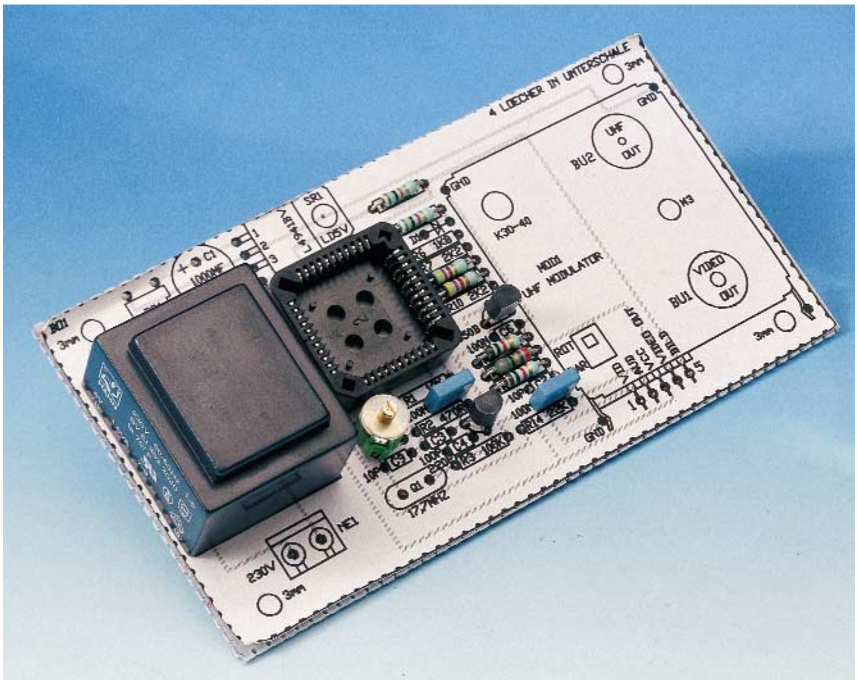


Figure 7. The 1:1 layout drawing is glued to the perforated prototyping board.

for TRACK, MULTILAYER for PAD and MIDLAYER 1 for mechanical elements. **Figure 6** shows a combined print of all four layers.

When the board layout is finished, you can use EASYPLOT to convert it into a plotter file (*.dft) and plot it. The plotter output provides the most options, since a different pen can be assigned to each layout layer. Each layer can be locked or flipped.

PRINTGL.ZIP

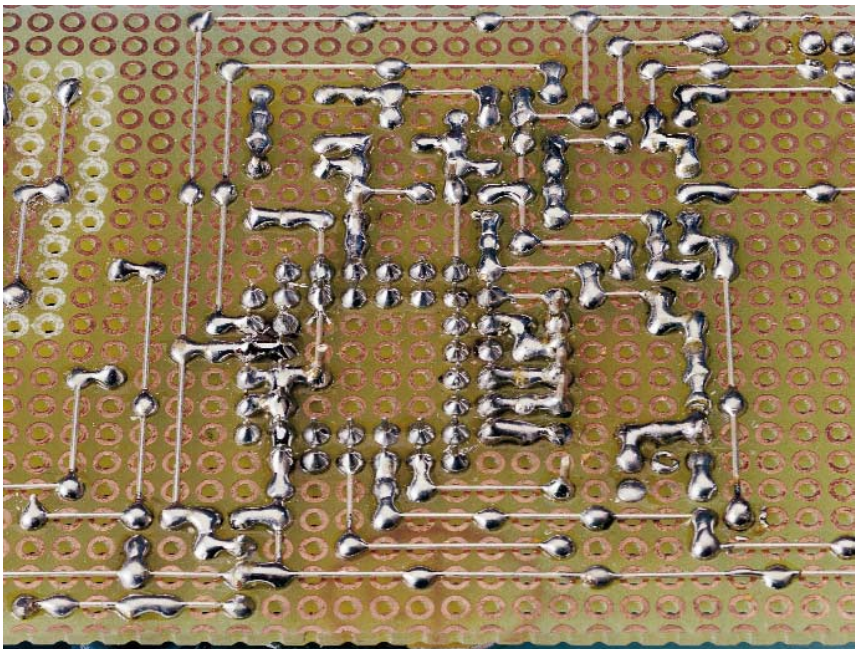


Figure 8. The components are first completely soldered…and the interconnecting wires are then attached.

If you do not have a plotter, you must convert the dft file into a print file. The files located in the PRINTGL.ZIP archive can be used for this. You will find plotter emulators for Windows, a printer driver for DOS-CAD and, once again, an extensive ReadMe file. The files in the zip archive are unpacked and directly copied to C:\PRINTGL.

The plotter program is called using the user interface PMI.EXE. The line width and density can be individually selected for each pen.

TVG.EXE

If *Easytrax* is used especially for this construction project, it is easier if you load the self-extracting file TVG.EXE (also available from the *Elektor Electronics* website) into the EASYTRAX directory and start it up before starting the program. This file contains the complete instructions, as well as the libraries and default settings. Any matching program modules that may already be present may be overwritten when the file is unpacked. TVGSCM.LIB is for the schematic diagram of the pattern generator, while TVGPCB.LIB is for the layout. Both libraries are located in TBG.EXE.

Solder-pad matrix

Now you can print the layout at 100% scale, cut it out and glue it onto the perforated prototyping board (see **Figure 7**). This spares you the tiresome task of counting holes. If possible, you should use an epoxy-glass prototyping board for reasons of stability, but a Pertinax board can also be used. The cutting lines always run through the middle of the holes. You can score a Pertinax board on both sides with a pointed tool and then break it, while an epoxy board can be readily cut with a heavy shears. You can leave your saw in the drawer.

The components are inserted through the holes as usual. Pull their leads through as you would with a 'real' circuit board, bend them over on the solder side, cut them short and then solder them (see **Figure 8**). Lead wires are not well suited for making tracks. Only after all the components have been mounted should you start adding the interconnecting wires. For this the board should be held level, preferably firmly secured in a bench vice. You will need tinned or silver-plated hookup wire with a diameter of approximately 0.5 mm for the interconnections. Pull several pieces around 30 cm long straight using two smooth-jawed pliers. In order to avoid excessive warping with long connections, due to thermal expansion during soldering, it is best not to bridge more than 4 to 5 holes at a time. The use of a small bending gauge is highly recommended, including for the wire bridges. In order to make a connection 'around the corner', you must use two wire bridges. Very short connections can be made using solder alone.

The 1:1 printout can also be used as a drilling template for the 25-mm deep upper shell of the enclosure. The circuit board is firmly fixed to the 15-mm deep lower shell on the modulator side, using four 10-mm long threaded standoffs with 3-mm internal threads. The Euro mains cable can be fitted with a plastic strain relief clamp of suitable diameter.

(000084-1)

Note:
The programmed EMP7064 for this project is available through our Readers Services as order code **000084-31**.

On the Internet

Easytrax (Protel):
http://www.protel.com/download/dos/easytrax.zip

Component library and German documentation:
http://www.ele.uri.edu/~mr/services/easytrax/

PRINTGL.ZIP plotter simulator:
http://www.concentric.net/~Ravitz/#df

Eagle Light (CADSOFT):
http://www.cadsoft.de

Target 2001!V8 discover (Ing.-Büro Friedrich):
http://www.ibfriedrich.com

All urls are available on the Hyperlinks page of the Elektor Electronics website, www.elektor-electronics.co.uk